Available online at http://www.journalcra.com

International Journal of Current Research
Vol. 10, Issue, 07, pp.71061-71066, July, 2018

INTERNATIONAL JOURNAL OF CURRENT RESEARCH

ISSN: 0975-833X

RESEARCH ARTICLE

NUMERICAL STUDY OF FLOW IN THE WATER INLET OF THE PENSTOCK OF A HYDROELECTRIC DAM

[1,]*Tchawe Tchawe Moukam, [2]Tcheukam-Toko Denis [3]Kenmeugnebienvenu and [4]Djiako Thomas

[1]PhD student, Department of Mechanical Engineering ENSAI, University of Ngaoundéré
[2]Associate Professor, Department of Mechanical Engineering COLTECH, University of Buea
[3]Associate Professor, Department of Mechanical Engineering HTTTC-Kumba, University of Buea
[4]Senior Lecturer, Director of the Master's School, University Institute of Guinea Gulf (UIG), Douala

ARTICLE INFO

Article History:
Received 25th April, 2018
Received in revised form
04th May, 2018
Accepted 19th June, 2018
Published online 30th July, 2018

Key words:
Water intake, Penstock,
Hydroelectric dam,
2D numerical modeling.

ABSTRACT

In This work is the result of research conducted on a dam in Cameroon. The aim is to study the variation of flow through the intake of a penstock in order to highlight its influence on the entire structure. For this fact, a study is conducted by a 2D numerical approach under FLUENT (6.3.26). It is clear from this work that the shape of the structure upstream and the intake have a real impact on the structure of the flow, it is the same for the flow rate of water that arrives upstream. The structure of the flow is unstable, the maximum speed is observed not far from the wall. A large recirculation zone is observed on the step at the inlet of the intake, it is very active on the acceleration of the flow velocity towards the pipe; likewise, it is responsible for the birth of air bubbles which will have consequences later on the phenomenon of cavitation of the blades of our turbine.

Copyright © 2018, Tchawe Tchawe Moukam et al. *This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.*

Citation: Tchawe Tchawe Moukam, Tcheukam-Toko Denis and Kenmeugne Bienvenue, 2018. "Numerical study of flow in the water inlet of the penstock of a hydroelectric DAM", *International Journal of Current Research*, 10, (7), 71061-71066.

INTRODUCTION

In a context where global electricity production is dominated by fossil fuels (about 60% of global electricity production according to OFEG and EDF quoted by Ketcheuzeu *et al.*, 2014), hydropower is a solution for the future. According to Mohammed and Rasim, Global production of hydropower-based electricity is about 20% of total electricity production. At a strangulation of the banks of a watercourse, a dam is erected which creates a reservoir of water. At the foot of this dam, turbines connected to alternators are installed (source ENEO, 2015). A convergent directs water under pressure to the pipe. This convergent, of varied form, is in the technical language called water intake. This area is the subject of many studies because its shape greatly influences the structure of the flow. We can among other things state the work of Islam *et al.* (2011) who shows that by they study the Schwarz-Christoffel transformation can be used to estimate the flow upstream of two-dimensional rectangular intakes having variable sizes and locations and for nozzle-shaped intakes.

Corresponding author:* **Tchawe Tchawe Moukam,
PhD student, Department of Mechanical Engineering ENSAI, University of Ngaoundéré
DOI: https://doi.org/10.24941/ijcr.31203.07.2018

It is shown that flow acceleration region depends on water depth, location of intake, and intake size. The location of the peak velocity can deviate away from the centerline of the intake. Montazerin *et al.,* (2014) presents a study on the flow structure and turbulence characteristics of a horizontal water intake. Meselhe *et al.,* (1998) developed a numerical flow model for evaluating different fish by pass systems at the Wanapum Dam on the Columbia River, Wash. But, the importance of this work is reflected in some other studies (Gerges and Mc Corquodale, 1997; Shammaa and Zhu, 2010) who agree on the fact thatanalyzing flow upstream of intakes is important in many engineering applications, e.g., fish entrainment study, flow in sedimentation tank, flow induced by sluice gates, skimmer wall, and temperature control curtain. When it comes to larger pipes, other approaches can be used; this is because the measurements are more difficult. C. Wang *et al.,* (2012) mainly conduct their study on accuracy of the ultrasonic flow meter used in the hydro turbine intake penstock of the Three Gorges Power Station. This is the case of many other studies (Li *et al,* 2008; Lai and Khan, 2011). Following the presence of the step upstream of our intake, studies on the presence of forward-facing step have been of great support. This is particularly the case of the works of Moss et Baker (1980) and Abdalla *et al.,* (2009). This study is a contribution to the understanding of the state of flow in the upstream zone and the intake in general; but its peculiarity lies on the model that is studied. Indeed, the shape of the structure and the boundary conditions are that of the biggest dam of production in Cameroon at the present time. The importance of this work is more focused on a local level.

MATERIALS AND METHODS

The material studied here is the Songloulou power station. The working area is that of the upstream part near the intake dam and the actual water intake. The figure below presents the mesh of the domain (without taking into account the penstock).

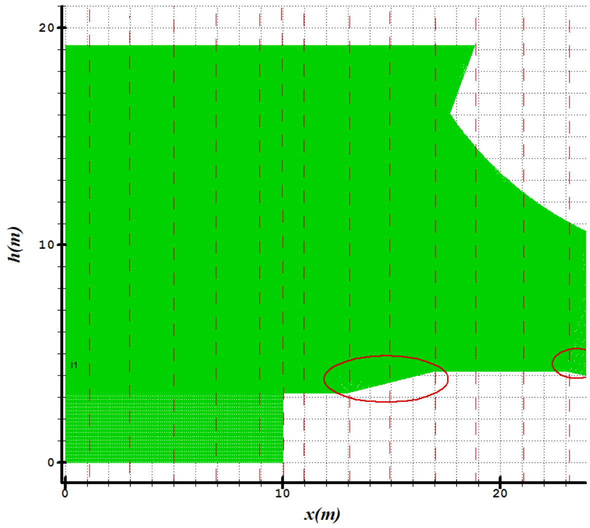


Figure 1. Mesh domaine

An emphasis will be placed on the red framed parts which are at respective distances x = 13m, x = 15m, x = 17m for the first part, and x = 23m for the second part. Standard parameters under Fluent are kept with model k-ε RNG for calculation. The upstream flow is assumed to be stationary. For the convective and the diffusive terms, a second order upwind method was used while the SIMPLE (Semi Implicit Method for Pressure Linked Equations) procedure was introduced for the velocity pressure coupled to a multiphase model (VOF) (Patankar quoted by Tchawe *et al.,* 2015). The convergence criterion was set at 10^{-8} to better refine our results.

Simplifying assumptions

Knowing that water is an incompressible fluid, we will impose some assumptions for our work:

- The turbulence is isotropic;
- The resistance of the air is negligible;
- The parietal effects are neglected (2D work).

General equation of the problem

Monophasic turbulent flows in a closed pipe are described by the Navier-Stokes equations. To these equations, we add the equation of turbulent kinetic energy and that of its dissipation as proposed by Launder and Spalding.

$$\frac{\partial(\rho \bar{u}_i)}{\partial x_j} = 0 \qquad (1)$$

$$\frac{\partial(\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial \bar{u}_i}{\partial x_i}\right)\right] + \frac{\partial(\rho \overline{u_i' u_j'})}{\partial x_j} + F_i \qquad (2)$$

Relationship (1) is the continuity equation, and (2) the quantity of motion conservation equation with:

- The left term represents convective transport;
- The first term on the right represents the forces due to pressure;
- The second term on the right represents the viscosity forces;
- The last two terms on the right represent the forces generated by the turbulence.

where ($\rho \overline{u_j' u_i'}$) is the Reynolds tensors

The closure problem is solved using the Boussinesq hypothesis:

$$-\rho \overline{u'_i u'_j} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}(\rho k)\delta_{ij} \qquad (3)$$

Dimensional analysis provides a value scale of large vortices and allows the viscosity to be related to the two variables k and ε . We show a coefficient of proportionality that is obtained experimentally:

$$L_t = c_\mu \frac{k^{3/2}}{\varepsilon} \qquad (4)$$

$$\upsilon_t = c_\mu \frac{k^2}{\varepsilon} \qquad (5)$$

To determine υ_t , it is necessary to calculate the two variables k and ε . The equations of turbulent kinetic energy and its dissipation rate give us:

$$\rho \bar{u}_j \frac{\partial k}{\partial x_j} = \frac{\mu_t}{\sigma_k}\frac{\partial^2 k}{\partial x_j^2} + \rho p_k - \rho\varepsilon \qquad (6)$$

$$\rho \bar{u}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\mu_t}{\sigma_\varepsilon}\frac{\partial^2 \varepsilon}{\partial x_j^2} + \rho\frac{\varepsilon}{k}(c_{\varepsilon 1} p_k - c_{\varepsilon 2}\varepsilon) \qquad (7)$$

These equations can be rewritten for turbulent kinetic energy in the form:

$$\bar{u}_j \frac{\partial k}{\partial x_j} = c_\mu \frac{k^2}{\varepsilon}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\frac{c_\mu k^2}{\sigma_k \varepsilon}\frac{\partial k}{\partial x_j}\right) - \varepsilon \qquad (8)$$

- The term on the left represents the variation of turbulent kinetic energy;
- The first term on the right represents the production of turbulent kinetic energy;
- The second term on the right represents diffusion;
- The last term of the right is dissipation.

The equation of dissipation energy is given by the equation:

$$\overline{u_j}\frac{\partial \varepsilon}{\partial x_j} = c_{\varepsilon 1} c_\mu k\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right)\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\frac{c_\mu k^2}{\sigma_\varepsilon \varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) - c_{\varepsilon 2}\frac{\varepsilon^2}{k} \quad (9)$$

Constants determined from elementary experiences are:

$$c_\mu = 0.09,\ \sigma_k = 1,\ \sigma_\varepsilon = 1.3,\ c_{\varepsilon 1} = 1.44,\ c_{\varepsilon 2} = 1.92 \quad (10)$$

Three mesh distributions have been tested to ensure that the calculated results are grid independent. The height of roughness has been introduced as a term k_s in relation with the Strickler coefficient K, calculated by the formula (11) bellow, according to Sinniger and Hager, [18].

$$K k_s^{1/6} = 8.2\sqrt{g} \quad (11)$$

RESULTS AND DISCUSSION

Boundary condition

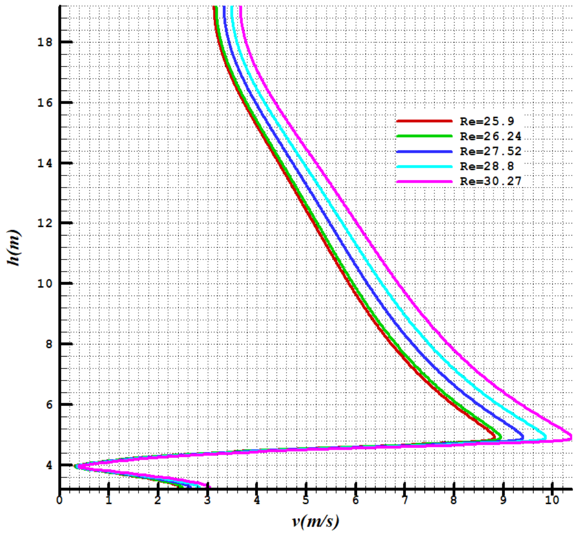
The level of water of the study took for this first work is the minimum level in this dam for a maximum power, provided in the document of the company guarantor of works of the energy field in the country. The variation of the Reynolds number of the upstream flow will be made according to 5 values, which will be further reduced to three according to the minimum, the intermediate and the maximum flow for maximum power (Re=25.9x10^6, Re=28.8x10^6, Re=30.27x10^6).

Upstream zone (close to intake)

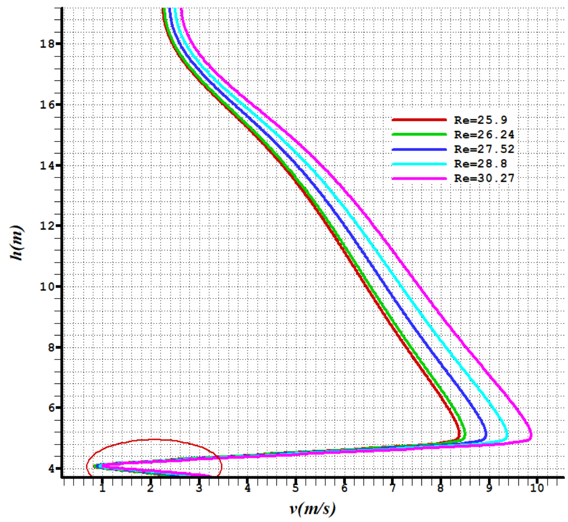
This part is characterized by a forward-facing step (which is the subject of another study), followed by a convergent slope (the big part circled in red), and a recovery zone (downstream of the slope). All previous elements coupled to the structure of the upper flank form what is called funnel (convergent), which in this case represents our water intake. The figure above (Figure 2-a) shows the behavior of the flow at the beginning of the slope (x = 13m). We note from study that it is the zone whose form of the flow near the lower wall is the most unstable. The recirculation phenomenon is maximum. This is due in particular to the sudden deviation of water, caused by the slope, which creates a comfort zone to animate the vortex structure in the flow at this level. In the same direction at the zone x = 15m (Figure 2-b), the structure (shape) of the flow is still very disturbed. However, we observe the decrease in the intensity of the vortex flow, which remains all the same very active. The latter (vortex flow) propagates along this slope and creates a detachment of the turbulent layer. In Figure 2-c, we observe a total disappearance of the vortex structure near the lower wall. It is characterized by the attachment of the turbulent layer on the wall, and the normal detachment of the turbulent boundary layer for a real fluid flow.

In the intake flow

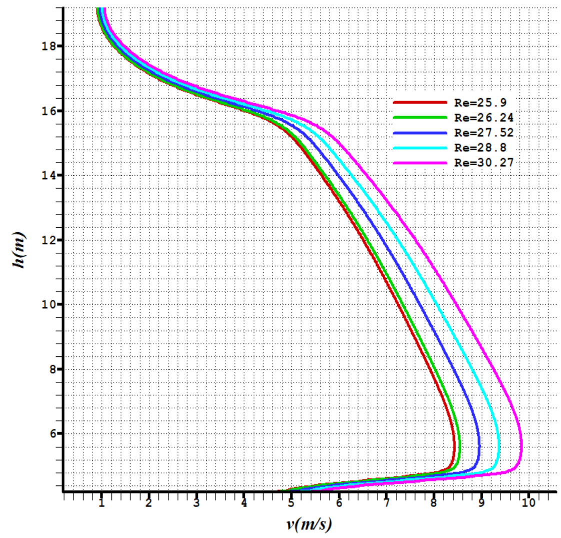
Figure 3-a is characterized by the acceleration of the velocity and whose peak is not far from the lower wall. In Figure 3-b, the finding is the same, but with a recovery of the structure of the flow in the axis of the intake which characterizes a slight deceleration of the fluid at this level in the pipe. We also note the latter is still stronger near the walls than the axis in this area. The work done by Wang *et al.* (2012) show similar results. Figure 3-c clearly shows the decelerating effect of the flow towards the axis of the conduit; but what catches our attention is the point where the peak of the velocity of the flow is situated.



(a)



(b)



(c)

Figure 2. Velocity profile for different Re(x10^6) à : (a) x=13m, (b) x=15m, (c) x=17m

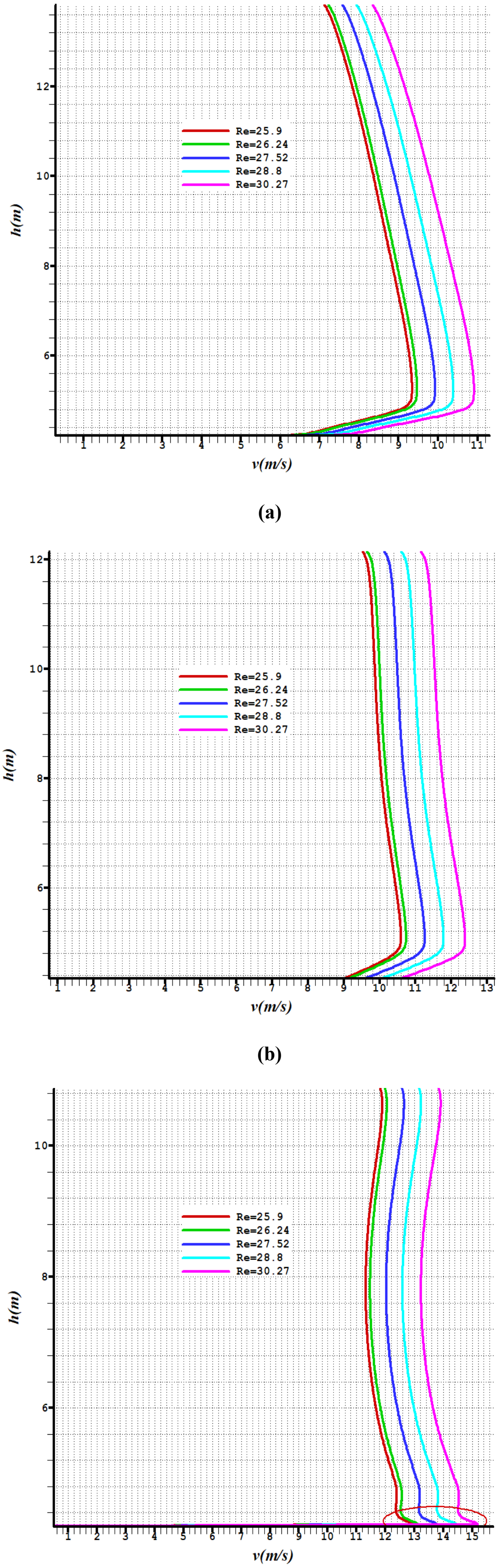


Figure 3. Velocity profile for different Re(x10[6]) à: (a) x=19m, (b) x=21m, (c) x=23m

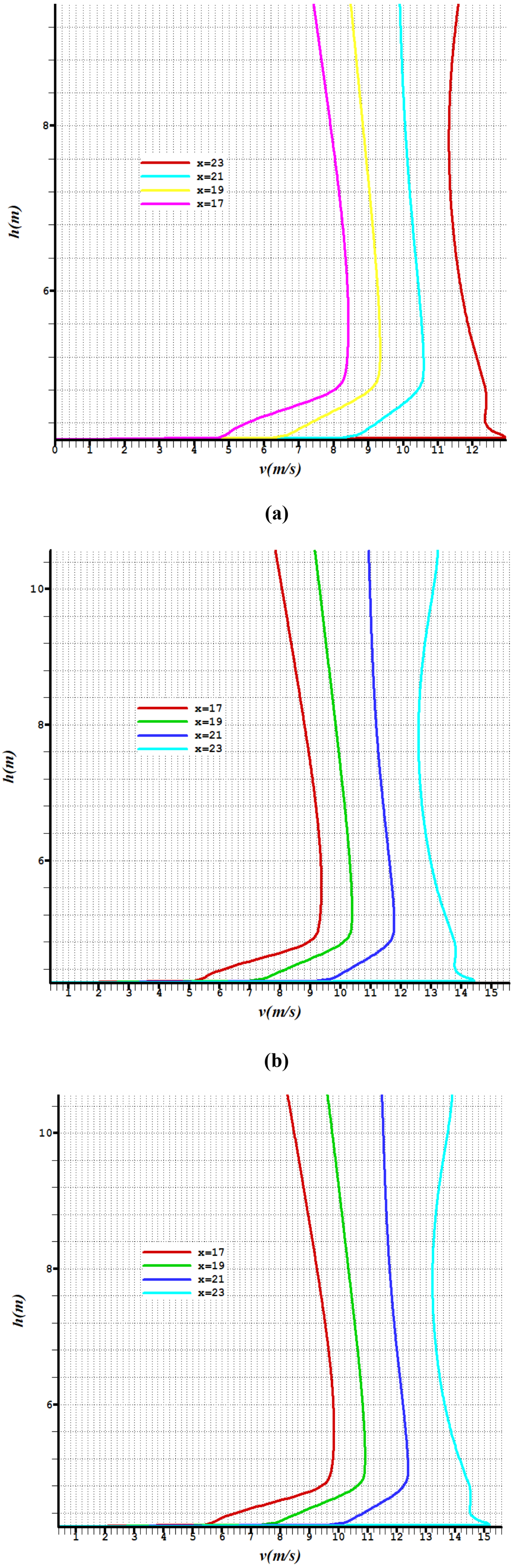


Figure 4. Velocity profile for different x position with: (a)Re=25.9x10[6], (b)Re=28.8x10[6], (c)Re=30.27x10[6].

It is so close to the wall that we can say that the roughness of our wall and the viscosity of water are zero at this point. Now, this is justified in our case simply by the fact that it is exactly this point that is the inclination that issues directly to the conduit (point of connection between intake and penstock); knowing that the conduit is inclined with respect to the horizontal.

Velocity profil for the three Reynolds numbers retained

Four (04) positions are considered to observe and summarize this study, with three (03) different Reynolds numbers as shown in the figures above. From the position x = 17 to position x = 21m, the detachment of the turbulent boundary layer is substantially normal to that of a fluid-structure interaction in a real case of the high-valued flow of D. The structure of the flow at position x = 23m is particular and is especially due to the inclination of the pipe.

However, the major remark to note is that whatever the flow rate of the upstream flow, the structure of the flow at these different positions remains the same. Likewise, the velocity of the flow increases the upstream flow rate.

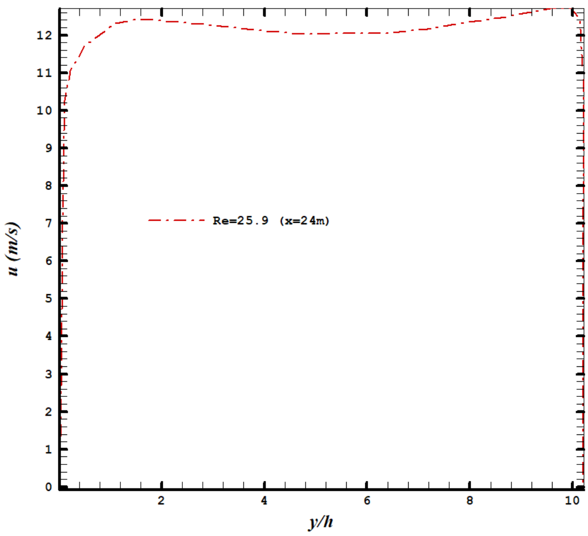


Figure 5. Velocity distribution for Re=25.9x10^6 , present study

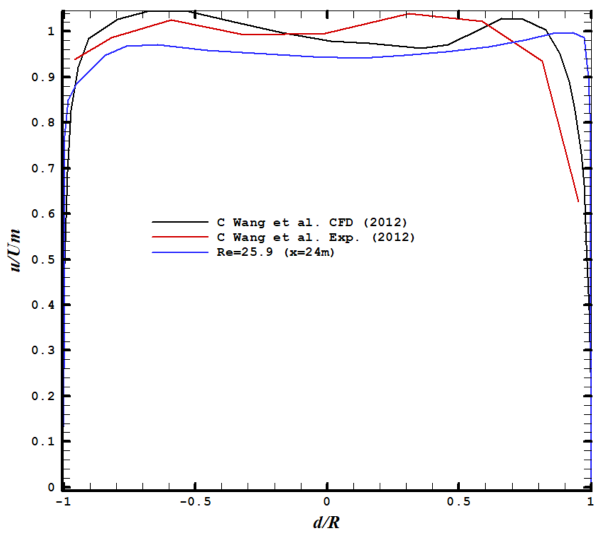
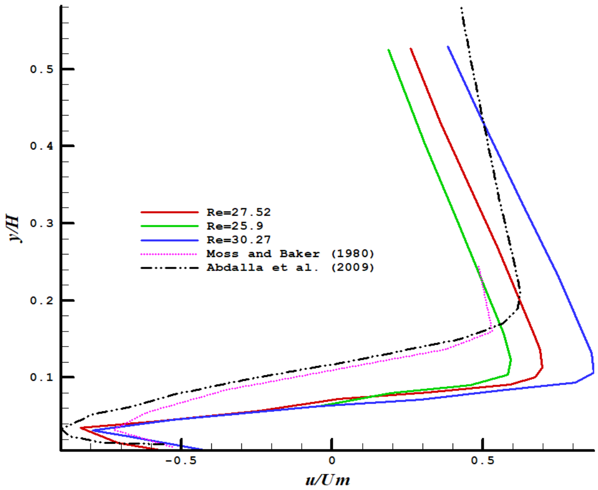


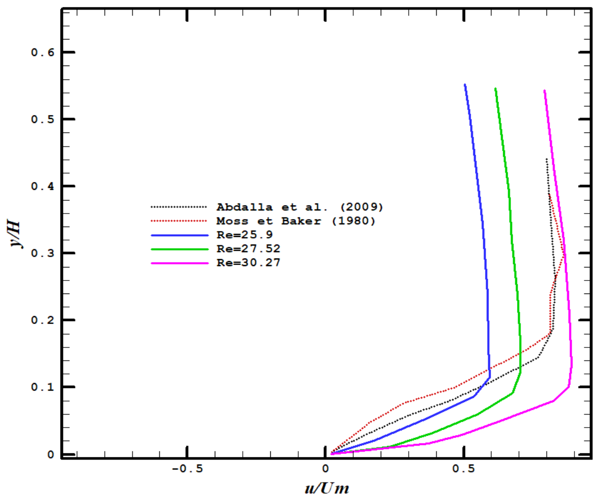
Figure 6. Velocity distribution for Re=25.9x10^6 (present study), C. Wang et al. CFD (black), C. Wang et al. Exp. (Red).

Comparative Study and Analysis

At a position x at the initiation of the penstock by the flow (x = 24m), we sought the structure of the flow in order to observe certain theoretical parameters. The results shows that contrary to the standard structure of the flow in a charged real fluid with small diameter where the velocity is maximum in the center of the pipe, this is still not the case in large diameter configurations like noted above. Similar results have been observed, such as those Wang *et al.* (2012) on theintake penstock of the Three Gorges Power Station. We clearly see that the structure of flow in a real context still does not follow the theoretical logic. Nevertheless, we can justify this behavior by some elements namely: the diameter of the conduit, the angle of inclination of the conduit (element on which advanced studies will be accentuated), the flow rate, just to name a fewsince the fluid here is water. Another study that gives us enough information is that conducted by Abdalla *et al.,* (2009) who sought the influence of a subsea obstacle and a step on the flow structure at the point of the obstacle and downstream. The similarity is that upstream of our water intake, we have a step. We therefore chose two positions of ***x*** downstream of the latter to evaluate the results (x = 15m and x = 19m in our case study).



(a)



(b)

Figure 7. Velocity profil: (a) at the position of x=15m with this study represented by Re=25.9x10^6, Re=27.52x10^6, Re=30.27x10^6 ; (b) at the position of x=19m with this study represented by Re=25.9x10^6, Re=27.52x10^6, Re=30.27x10^6

We notice the same trend in the flow structure at this level. The deviation observed is particularly due to the fact that their configuration was a completely free surface flow, while ours is close to a transition zone (from a free surface flow to a flow in closed conduit). These profiles explain in a way the unsteady behavior of the flow in this part of the dam.

Conclusion

The water intake of the Songloulou dam is special, from the point of view of the elements that surround it. Unlike others that open directly on the upstream slope, it is preceded by a step which naturally has a considerable impact on the structure of the flow that is heading towards the conduit. First, a strong recirculation upstream of the forward-facing step, then an acceleration of the flow velocity not far from the lower wall towards the throttling zone. A throttling zone that, like the upstream step, hosts another recirculation zone. This boosts once again the flow velocity of water towards the conduit with a significant detachment of the turbulent layer. The flow in this zone is particularly unsteady.

Acknowledgment

Special thanks to the School/Management of Masters of the University Institute of the Gulf of Guinea (IUG) of Douala-Cameroon for the enormous support given for the realization of these studies.

REFERENCES

Chi Wang, Tao Meng, He-ming Hu, Liang Zhang, 2012. "Accuracy of the ultrasonic flow meter used in the hydroturbine intake penstock of the Three Gorges Power Station", Flow Measurement and Instrumentation, 25 32–39.

Ibrahim Elrayah Abdalla, Zhiyin Yang and Malcolm Cook, 2009. "Computational analysis and flow structure of a transitional separated-reattached flow over a surface mounted obstacle and a forward-facing step. *International Journal of Computational Fluid Dynamics,* Vol. 23, No. 1, January 2009, 25–57.

Ketcheuzeu-Ngakam, J., Koueni-Toko, C., Djeumako, B., Tcheukam-Toko, D., Kuitche A. 2014. Characterization of hydrodynamics parameters of a Kaplan turbine. Energy and Power (EP), Vol. 4, issue 3, pp. 58 – 69.

Lai W. and Khan, A. A. 2011. A Discontinuous Galerkin Method for Two-Dimensional Shock Wave Modeling. Modelling and Simulation in Engineering Volume 2011, Article ID 782832, 10 pages.

Launder B. E. and Spalding, D. 1974. The numerical computation of turbulent flow computational methods. Applied. *Mechanical Engineering,* Vol. 3, pp. 269-289.

Li, W. P., Li, G. X., Li, H., and Zhang, K. W. 2008. Numerical simulation of water body resonance in penstock of hydropower station. *Journal of Hydraulic Engineering,* 39(1): 109–114.

MarghzarSh, H, Montazerin, N and Rahimzadeh, H. 2003. Flow field, turbulence and critical condition at a horizontal water intake. *Proc. Instn Mech. Engrs.,* Vol. 217 Part A: J. Power and Energy.

Mohammed, T. G. and Rasim, A. K., *Hydro Power* Chapter 4, 30pages. http://dx.doi.org/10.5772/52269

Moss, W.D. and Baker, S. 1980. Re-circulating Flows Associated with Two-dimensional Steps. Aeronautics Quarterly, 151–172.

PatankarS.V. 1980. Numerical heat transfer and fluid flow. McGraw Hill, New-York.

Rashedul Islam Md and David Z. Zhu, M. 2011. Flow Upstream of Two-Dimensional Intakes. *Journal of Hydraulic Engineering,* Vol. 137, No. 1.

Sinniger R. O. and Hager W. H. 1989. Constructions hydrauliques. Traité de Génie Civil, Vol. 15, Presses Polytechniques et Universitaires Romandes, Lausanne, Switzerland.

Tchawe, M., Djeumako, B., Koueni-Toko, C., Tcheukam-Toko, D. and Kuitche, A. 2015. Study of Dynamic Field around a Vertical Circular Cylinder placed in an Open-Chanel Flow. *International Journal of Innovative Science, Engineering & Technology,* Vol. 2 Issue 2.

Tcheukam-Toko, D., Mokem-Chetchueng, M., Mouangué, R., Beda, T., Murzyn, F.2013. Characterization of Hydraulic Jump over an Obstacle in Open-channel Flow. *International Journal of Hydraulic Engineering (IJHE),* 2(5), 2013, 71-84.

Wilson, D. 1998. Turbulence modeling for CFD. DCV industries, 2nd Edition.

Yaser Shammaa, David Z. Zhu, and Nallamuthu Rajaratnam, 2005. Flow Upstream of Orifices and Sluice Gates. *Journal of Hydraulic Engineering,* Vol. 131, No. 2.

Meselhe, E. A. and Odgaard, A. J. 1998. 3D numerical flow model for fish diversion studies at Wanapum dam. *Journal of Hydraulic Engineering,* Vol.124, No.12.

Eneo Cameroon S.A. Fluent, 2006. User manual 6.3.26

****